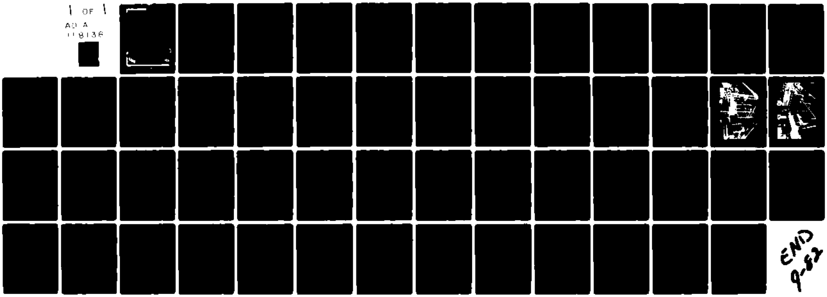
AD-A118 136 ARIZONA STATE UNIV TEMPE DEPT OF MECHANICAL AND AER--ETC F/G 20/4
TURBULENT SWIRLING FLOW DOWNSTREAM OF AN ABRUPT PIPE EXPANSION --ETC(U)
JUL 82 D E METZGER, G P NEITZEL N00014-81-K-0428
UNCLASSIFIED CR-R82036 NL

1 OF 1
AD A
118136



END
9-82

(12)

mechanical and aerospace engineering

mechanical engineering, aerospace engineering, energy systems engineering, engineering science, material science

Turbulent Swirling Flow Downstream of an Abrupt Pipe Expansion — Modeling and Experimental Measurements

Interim Progress Report
June 1, 1981 - May 31, 1982

Darryl E. Metzger
G. Paul Neitzel

Sponsored by the Office of Naval Research,
Contract No. N0014-81-K-0428, NR097-451

July, 1982

College of Engineering & Applied Sciences
Arizona State University
Tempe, Arizona 85287

CR-R82036

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER CR-R82036	2. GOVT ACCESSION NO. AD-A118136	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) TURBULENT SWIRLING FLOW DOWNSTREAM OF AN ABRUPT PIPE EXPANSION -- MODELING AND EXPERIMENTAL MEASUREMENTS		5. TYPE OF REPORT & PERIOD COVERED Interim 1 Jun 81 - 31 May 82
		6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) Darryl E. Metzger, G. Paul Neitzel		8. CONTRACT OR GRANT NUMBER(s) N00014-81-K-0428 NR097-451
9. PERFORMING ORGANIZATION NAME AND ADDRESS Dept of Mechanical and Aerospace Engineering, Arizona State University, Tempe, AZ 85287		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
11. CONTROLLING OFFICE NAME AND ADDRESS Office of Naval Research, Power Program, Arlington, VA 22217		12. REPORT DATE July 1982
		13. NUMBER OF PAGES 52
14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office) Dept. of the Navy, ONR Resident Representative, Bandelier Hall West, University of New Mexico, Albuquerque, NM 87131		15. SECURITY CLASS. (of this report) Unclassified
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

DTIC

AUG 1 1 1982

H

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

turbulent flow, heat transfer, swirling flow, sudden expansion.

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

This report covers activities during the first year under ONR contract N00014-81-K-0428. The project is a combined experimental and numerical investigation of heat transfer and fluid dynamic characteristics of a turbulent swirling flow in a pipe downstream of a sudden expansion. Local wall heat transfer measurements as well as detailed flowfield measurements utilizing laser-Doppler velocimetry (LDV) techniques are planned for the experiments. Numerical modeling efforts will focus on the development of a code which is

DD FORM 1473, 1 JAN 73 EDITION OF 1 NOV 65 IS OBSOLETE

S/N 0102-LF-014-6601

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

20. ABSTRACT (continued)

which is efficient enough to be useful for design purposes, while accurately modeling the flow field.

First year efforts have concentrated on the preliminary design considerations for the experimental facilities and, subsequently, on detail design and construction of the individual facility components. The LDV system suitable for the planned tasks has been specified, purchased, and used on simple flows for familiarization purposes. Preliminary numerical work has concentrated on exploring the capabilities of the TEACH code in order to assess its suitability for the complex turbulent flows generated by the combination of abrupt expansion and swirl.

TABLE OF CONTENTS

I. INTRODUCTION

The objective of the program is to address the problem of flow and heat transfer in regions following a sudden enlargement of a circular pipe carrying a swirling flow. Previously, effects of both swirl and sudden expansion on heat transfer have been investigated separately, and each is found individually to cause very significant elevation of heat transfer over normal non-swirling, constant area flows. The elevation is a strong function of axial location downstream of the expansion or swirl producing locations, with gradual restoration to normal fully developed pipe flow values.

Little information exists in the literature to help the designer or analyst assess the effects when both swirl and sudden expansion occur together. It is obvious that the combined behavior will not be predictable, in general, from knowledge only of the two effects separately. The presence of swirl is likely to have a strong influence on the location of reattachment following the expansion, and probably also on the magnitude of the reattachment point heat transfer rate. The presence of expansion with swirl will probably result in flow reversal along the pipe centerline over at least some of the parameter ranges of interest. The rate at which normal fully developed conditions are approached in the downstream region may be quite different than corresponding rates for either expansion or swirl alone.

The overall approach chosen for the investigation is a combination and coupling of both experimental and numerical work. It is intended that this work will illuminate the fundamental character of the flow and also provide accurate measurements of wall heat transfer rates. The experiments are to be well controlled and documented, and carefully conducted so that the results may be confidently used to judge the accuracy of the numerical modeling efforts.

In addition to these fundamental objectives, efforts will be made to interpret and cast the results into a form useful in design. The designer may be faced with a thermal problem as a result of the elevation and/or variation of heat transfer rates around the abrupt expansion. The state of the art in numerical models for turbulent flows of this complexity does not at present permit their use alone for confident design predictions of the wall heat transfer rates. As a result, in the work planned for this project, the measured heat transfer results will be interpreted and correlated in terms of the measured flow characteristics. In parallel, the flowfield measurements and flow modeling efforts will be directed at providing a validated model that can predict overall flow features pertinent to the surface heat transfer rate, such as location of reattachment, existence of on-axis recirculation, and tangential velocity component decay. If successful, the heat transfer correlations, together with a validated flow prediction code, should constitute a very useful tool for the designer.

II. PRELIMINARY DESIGN CONSIDERATIONS

A. Test Section Size

Preliminary work concentrated on establishing the general size of the test section. It was decided to design the flow facility so that a range of area expansion ratios could be accommodated for each of three different inlet pipe sizes. The expansion ratios and Reynolds numbers for each size should overlap with the other sizes so that confirmation of the geometric non-dimensionalization of results could be obtained.

These requirements, along with other contraints, dictated the final choice of the upstream and downstream pipe sizes. The other constraints included the necessity for a large enough pipe size to give good resolution for the LDV measurements, the total electrical power available for test section heating, cooling water capacity for test loop temperature stabilization, and pump, heat exchanger, power supply, and pipe component costs.

The above considerations were also interrelated with the choice of a method of providing the necessary electrical heating to the test section. Three methods of providing this heating were explored: (i) ohmic heating of thin test section walls by passing current directly through them, (ii) ohmic heating of thicker test section walls (segmented) by means of attached resistance heaters, and (iii) steam heating of jacketed test section walls. Choice (iii) establishes a uniform wall temperature thermal boundary condition but poses significant difficulties in resolving local heat flux values. Choice (ii) potentially allows establishment of an arbitrary wall thermal boundary condition but poses significant difficulties in achieving the heat addition levels necessary for good temperature difference resolution with the high Reynolds number flows with water as the working fluid. Ultimately, ohmic heating of thin test section walls was selected and used to help size the test section.

B. Thermal Boundary Conditions

As part of the choice of the test section heating method, consideration was given to the subject of thermal boundary conditions in general. These considerations, along with the decision to use current-carrying thin wall stainless steel tubes, resulted in a choice of thermal boundary condition to be used in the heat transfer test work, with variations possible for future work. The following paragraphs give an overview of the thinking which went into the selection.

For internal flows, it is customary for heat transfer coefficients to be based on the difference between the local surface temperature, $t_s(x)$, and the local bulk average or mixed near temperature, $t_m(x)$. Here x is the coordinate long the pipe axis. The mixed mean temperature is a useful reference temperature because it is usually easy to calculate in both experiments and applications from a simple energy balance.

In simple internal convection situations, the surface heat flux can be uniquely related to the two temperatures t_s and t_m and is thus a two-temperature or two-condition convection situation. However, there are many situations where more temperatures are required to specify the heat flux. For example, internal flow in an annulus with unequal temperatures on the inner and outer walls is a three-temperature problem which has been examined in detail.

In reality, convection problems are in general multi-temperature problems. In particular, it appears that the present problem of swirling flow in an abrupt expansion may be one where the multi-temperature nature is important. In an actual application with the swirl and sudden expansion, it is likely that neither heat flux nor surface temperature is uniform in the axial direction. The heat flux at some location in the region of interest downstream of the expansion may depend on $t_s(x)$ both upstream and downstream of the expansion. It appears very likely that the upstream surface temperature, at least, can have a significant influence on the downstream region heat transfer rates. Nevertheless, many of the previous studies of sudden expansion without swirl have used adiabatic upstream conditions and applied wall heating only downstream of the expansion. In order to provide comparison with previous work, and to begin heat transfer testing with the least complicated case, it was decided to first fabricate and test with heating applied only to the wall downstream of the expansion. Future work will attempt to acquire data with upstream and expansion face heating as well.

C. Transparent Test Section

Since flow field measurements are to be made using a laser-Doppler velocimeter (LDV), the test section to be used for this phase of the experimental work must be optically transparent to allow the laser beams to enter the flow field and scattered light to be collected. A transparent test section will also allow us to employ flow visualization in the preliminary stages of the work to see how gross flow properties (e.g., reattachment length and on-axis recirculation) are affected by Reynolds no. and swirl. The most likely candidates for test section material are precision glass and plexiglas tubing. Both have advantages and disadvantages. Both are difficult to obtain with uniform wall thickness and/or sufficient length for our purposes. Non-uniformities in wall thickness create problems with the accurate positioning of the LDV probe volume within the flow field and can even cause problems getting the beams to cross at all. Plexiglas has problems with crazing, resulting in a loss of transparency, while glass is subject to breakage. It appears that the most likely solution is to fabricate the test section in sections, using machined and polished plexiglas cylinders, and then to join these together to obtain the desired length.

Another problem pertaining to the use of the LDV is the fact that the geometry of the test section is cylindrical. This does not create serious problems when measuring either axial or tangential velocity components due to the way the beams enter the cylinder, but it does cause problems when attempting to measure the radial velocity component. One measure employed by many is to encase the portion of the test section of interest in a transparent box with planar walls and then to fill the gap with a liquid whose index of refraction matches that of the test section/box material. This eliminates the problem with the passage of the beams through the outer surface of the cylinder, but does nothing about the discontinuity at the cylinder/flow field interface if the flow field index of refraction does not match that of the cylinder. Another technique is to remove a portion of the cylindrical section and replace it with a thin film of a material like Mylar. This has essentially the same effect as the box method. For flow measurements in air, however, this has some nice advantages.

After examining the various options, the decision was made to not employ any such devices at least initially, but rather to allow the beams to enter the cylindrical surface and then use ray tracing techniques [1] to correct for the position and orientation of the sampling volume. In fact, once a reference location is established within the flow field, one can solve the inverse problem to determine the traverse required to effect a desired movement of the sampling volume. Should this technique prove troublesome or inadequate, then more exotic alternatives can be examined.

III. DETAIL DESIGN

A. Material Selection

An important consideration in the design of the test loop was that it would remain a permanent test facility in the thermosciences lab after the proposed work is completed. With this in mind, stainless steel was selected as the loop material for its high corrosion resistance and overall quality. In addition to stainless steel piping, other loop components and fittings were designed or purchased with all wetted parts stainless steel. The objective of building a permanent test facility required that the loop be designed on a very general basis, providing adaptability to a variety of test geometries and flow conditions. Thus, the test sections must be easily changed and the flow field must range from zero swirl to a high degree of swirl over a wide range of flow rates. These basic requirements guided the detailed design of the test loop.

B. Test Section Orientation

The first problem encountered in the loop design was whether the orientation of the test section should be horizontal or vertical. It appears from a study of the literature that natural convection may influence the flowfield over at least part of the operating range of interest. With the test section mounted horizontally, this effect could conceivably cause an asymmetry in the wall heat transfer. However, with thermocouples located circumferentially about the test section, buoyancy effects could be detected. With the test section oriented vertically, the buoyant convection would result in a symmetric effect on the heat transfer. The magnitude of the buoyant convection and the location at which it becomes significant would then be difficult to measure. It was also recognized that the test loop will require five feet (20 diameters of 3'' pipe) of straight developing pipe upstream of the test section. The combination of this developing length and the test section results in a straight run of about nine feet. With a vertical test section, this would require LDV measurements to be made as high as nine feet above floor level. The position stability required to made accurate LDV measurements would be difficult to maintain at such a height. For these reasons a horizontal test section orientation was selected. With the horizontal test section and circumferentially local measurements, the extent of the influence of gravity-induced convection can be determined. This is thought to be important, since swirling expansion flows in practice are found in both horizontal and vertical orientation. For future work in the project, however, asymmetric buoyancy may have to be eliminated. Therefore, the loop was designed so that with minimal modifications a vertical test section could be employed.

C. Specification of Test Section Tubing-Heat Transfer Measurements

In studying previous work done with flow past an abrupt expansion, it was estimated that 6 diameters of heated section upstream and downstream of the expansion would be adequate to study the flow features of interest for the swirling flow past a sudden expansion. Based on a largest potential expansion ratio of 3'' to 6'', the available test section length must thus be 54''.

In order to incorporate resistance heating for the heat transfer tests, the test sections are to be constructed of thin-walled stainless steel tubing. The initial experimental work will use standard tube sizes, both seamless and seam welded. These tubes are readily available and provide the least expensive and simplest starting point for the heat transfer measurements. Future experiments may use custom designed and built tubes.

The power supply to be used for heating the test sections was sized based on calculations using the thinnest available commercial tubing in the diameters of interest. These thin-walled tubes result in high resistances and potentially high heating capabilities. The size of the power supply was also limited by the 100KW available for heating. A D-C power supply was chosen to eliminate the potential problems associated with the wall thermocouples. With these guiding constraints, a D-C power supply rated at 6000A and 18V was purchased from the Rapid Electric Company.

The choice of direct resistance heating of the test section tubes to achieve a specified heat flux boundary condition dictates that tube sections with thin walls be used in order to provide the required electrical resistance. In order to further insure that the heat flux be uniform, the wall thickness itself must be quite uniform in both the circumferential and axial directions. This requirement, together with the relatively large test section tube diameters as specified in order to achieve good resolution in measured velocity quantities, presents a significant problem for the fabrication of the tubes. It also means that the strength of the tubes must be considered, with respect both to internal pressure capability and rigidity.

Several alternative methods of fabrication were explored with machine shop personnel and vendors. It was finally decided to use commercially available tubing.

With this decision, it is not possible to get exactly the inside diameters desired. Rather, the ideal test section is sized, based on the nominal diameter, and then the commercially available tube that most nearly matches the size of the nominal tube is selected. In sizing the tube, the most critical dimension is the tube wall thickness since the electrical resistance of the tube is a function of this. In order to utilize the entire 100KW available for heating, the electrical resistance of each test section must be set according to the simple power equation, $P = I^2R$. Since the maximum power and current (6000A) are known, the required resistance is calculated to be 0.003 ohm. With a resistance no larger than this value, the maximum possible heating will be attainable.

Assuming that the electrical resistivity of the stainless steel test sections remains constant, the total electrical resistance is

given by $R = \frac{\rho l}{A}$ (ρ = resistivity, l = length, A = cross-sectional area). Through simple manipulations using $R = P/I^2$ and $l = nD$ (n = number of diameters, D = nominal diameter) one obtains $R = \frac{\rho n}{\pi t}$ and then $t = \frac{\rho I^2}{\pi^2 q D^2}$, where q is the heat flux. This relation is for tube wall thickness of a straight tube test section, but the expansion test sections actually have three parts -- upstream, downstream and expansion plate. (For downstream heating only, the tube wall thickness is calculated assuming total heating. Then the same tubes can be used when total heating is desired.) Assuming the expansion plate to have negligible resistance, the total resistance of the expansion test section is given by, $R_{TOT} = \frac{\rho}{\pi}\left(\frac{n_{up}}{t_{up}} + \frac{n_{dn}}{t_{dn}}\right)$.
From the above expression for the thickness,

$$\frac{t_{up}}{t_{dn}} = \frac{D_{dn}^2}{D_{up}^2} .$$

Solving for t_{dn} in R_{TOT},

$$t_{dn} = \frac{\rho}{\pi}\left[\left(\frac{D_{up}}{D_{dn}}\right)^2 n_{up} + n_{dn}\right]\frac{1}{R_{TOT}}$$

Upstream and downstream lengths of 12'' and 42'' respectively were chosen for initial tests with a 2:1 expansion ratio and upstream tube diameters of 1 and 2 inches. This will give more than adequate length for the flow to redevelop and be completely studied. Using these lengths and the nominal tube diameters, the computed ideal wall thicknesses are given in Table III.1. The actual tubes purchased are also listed.

Nominal Tube sizes, inches (upstream/downstream)	IDEAL		PURCHASED				
	Upstream wall, inches	Downstream wall, inches	Upstream wall, inches	Upstream ID, inches	Downstream wall, inches	Downstream ID, inches	Actual Expansion Ratio
1/1	.126	.126	.120	1.010	.120	1.010	1.0
1/2	.288	.072	.120	1.010	.065	1.995	1.975
2/2	.252	.063	.065	1.995	.065	1.995	1.0
2/4	.144	.036	.065	1.995	.035	3.930	1.970

Table III.1 Tube Wall Thicknesses for Downstream Heating Only

It is expected that the thin-walled test sections may be the weakest structural component of the test rig. The pressure handling capability of these tubes was calculated using an empirically based bursting formula for thin-walled tubes. For the purchased tubes, these bursting pressure estimates are given in Table III.2 below.

INSIDE DIAMETER, INCHES	WALL THICKNESS, INCHES	OUTSIDE DIAMETER, INCHES	BURSTING PRESSURE, PSI (SAFETY FACTOR OF 4)
1.010	.120	1.250	3800
1.995	.065	2.125	1200
3.930	.035	4.000	350

Table III.2 Bursting Pressures for Test Section Tubing

As can be seen, for the planned initial tests, the tube strengths are more than adequate. However, the bursting pressure rapidly decreases as tube size increases, so that future tests with 6'' tubes may have to be restricted to low pressures.

D. <u>Heat Exchanger, Pump, Swirl Producer and Miscellaneous Items</u>

In passing through the heat transfer test sections, it is expected that the water will be heated to as much as 200°F. Since the system is a closed loop this water will be used again, but must be cooled before going back through the test section. This cooling is provided by a shell and tube heat exchanger with city water being the cooling fluid. This heat exchanger was sized by and purchased from American Standard according to specifications provided to them.

The potential Reynolds number operating range, particularly maximum Reynolds numbers, is controlled by the power available for heating the test sections. Using a Dittus-Bolter relation with a heat transfer magnification estimate of ten times that of non-swirling turbulent flow and with the known tube diameters, the maximum Reynolds numbers (based on upstream diameter) were estimated for the expansion ratio range planned for study and are listed in Table III.3. It was calculated that a water flow rate of 50 gallons per minute (gpm) would adequately encompass the desired range of Reynolds numbers. Based on this 50 gpm flow rate, 1 1/2'' schedule 10 type 304 pipe was selected for the loop. This pipe provides more than enough strength for future loop pressurization up to 150 psi. A conservative estimate of the pressure drop around the loop for the 1 1/2'' pipe was made in order to determine the pump requirements. The pump selected is manufactured by Ingersoll-Rand with a 75 psig head, 7 1/2 HP motor and rated at 50 gpm.

		Downstream Pipe Size				
		2''	3''	4''	5''	6''
Upstream Pipe Size	1''	1.67×10^5	6.94×10^4	3.75×10^4	2.24×10^4	1.44×10^4
	2''	2.2×10^5	1.17×10^5	7.12×10^4	4.54×10^4	3.04×10^4
	3''	--	1.30×10^5	8.92×10^4	6.16×10^4	4.36×10^4

($T_{WALL} - T_{WATER} \geq 20°$ FOR ENTIRE TEST SECTION)

TABLE III.3 Maximum Reynolds Number
(Test section lengths assumed to be 6 diameters for both upstream and downstream sections)

Several different alternative methods were considered for providing swirl to the flow upstream of the expansion. The source and nature of the swirl to be encountered in practical applications of the results of the project will vary. For this reason it is important to have a swirl generator in the test apparatus that is capable of providing, with relative ease, a wide range of swirl strengths. The method which appears to best fit this requirement at present is tangential injection. The final design decided upon consists of an 8'' diameter by 6'' long plenum with interchangeable plexiglas pipe inserts. The inserts contain four tangential slots, 1/8'' wide by 1'' long. A separately controllable secondary flow enters the plenum and passes through the tangential slots of the insert imparting a swirl component to the main axial flow. The axial flow enters the end of the insert from the upstream entry pipe. A sketch of the swirl producer is shown in Fig. III. 1. The unit is designed so that 1'', 2'' and 3'' (upstream tube diameters) inserts with 1/2'' wall thickness can be easily interchanged in the single plenum. Globe valves are located in the axial and tangential pipe lines leading to the swirl producer. With these independent controls, any flow ratio from entirely axial to entirely tangential entry flow can be established.

The flow rate through the swirl producer is monitored with two, 1'' turbine flowmeters manufactured by Flow Technology, Inc. One flowmeter is located in the tangential entry line to the swirl producer. The other is located in the main return line downstream of the test section. With the two meters, the flow rate through both

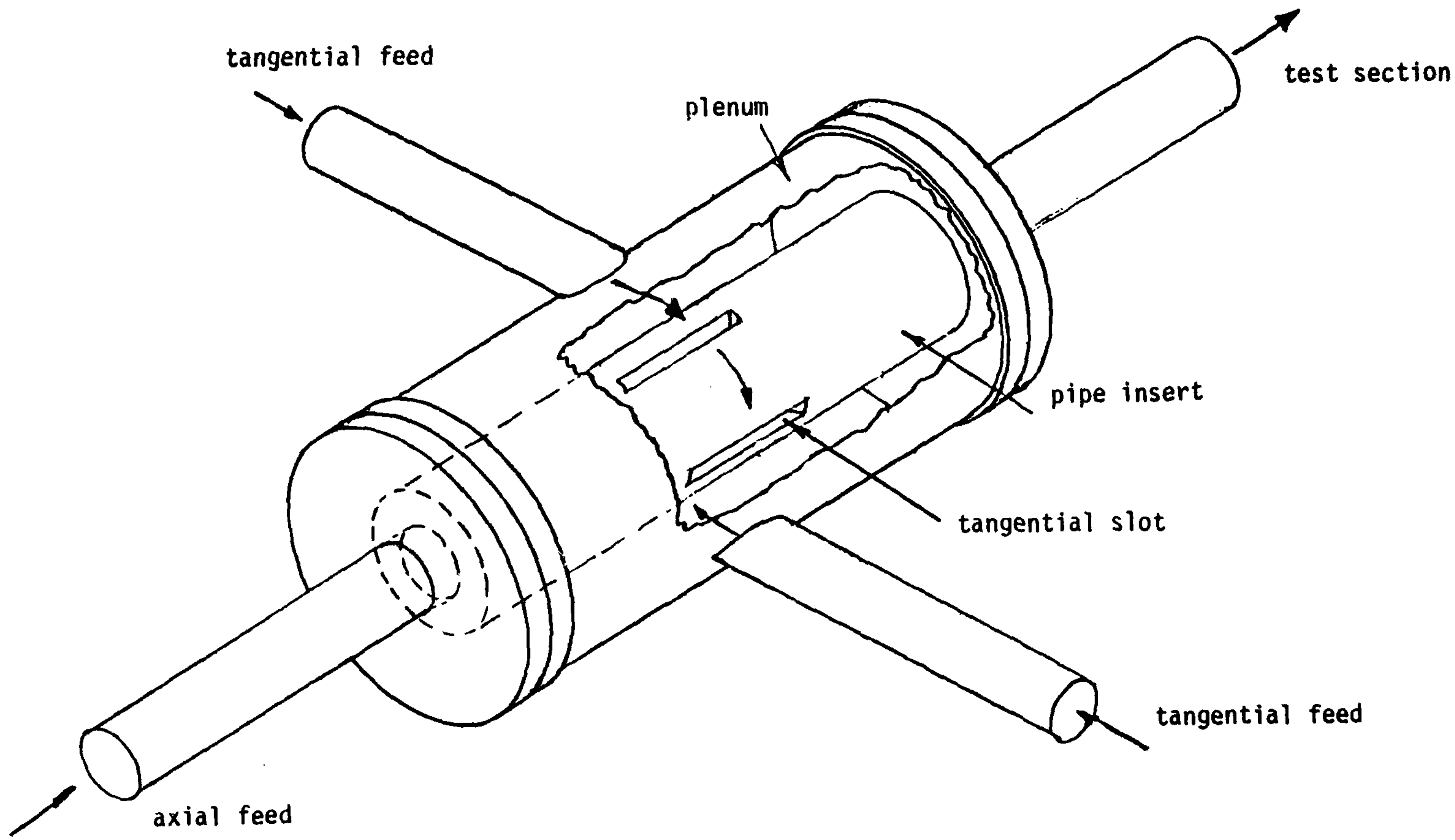


Fig. III.1 Schematic of swirl producer.

entry lines can be measured so that reproducing desired flow conditions is easily accomplished. Turbine flowmeters were chosen over venturies and orifices because of their high accuracy ($\pm$1%) over the entire 0 to 50 gpm flow rate range. Venturies and/or orifices cannot be easily specified to handle the entire desired range of flow rates. The turbine flowmeters also have the added advantage of direct digital readout.

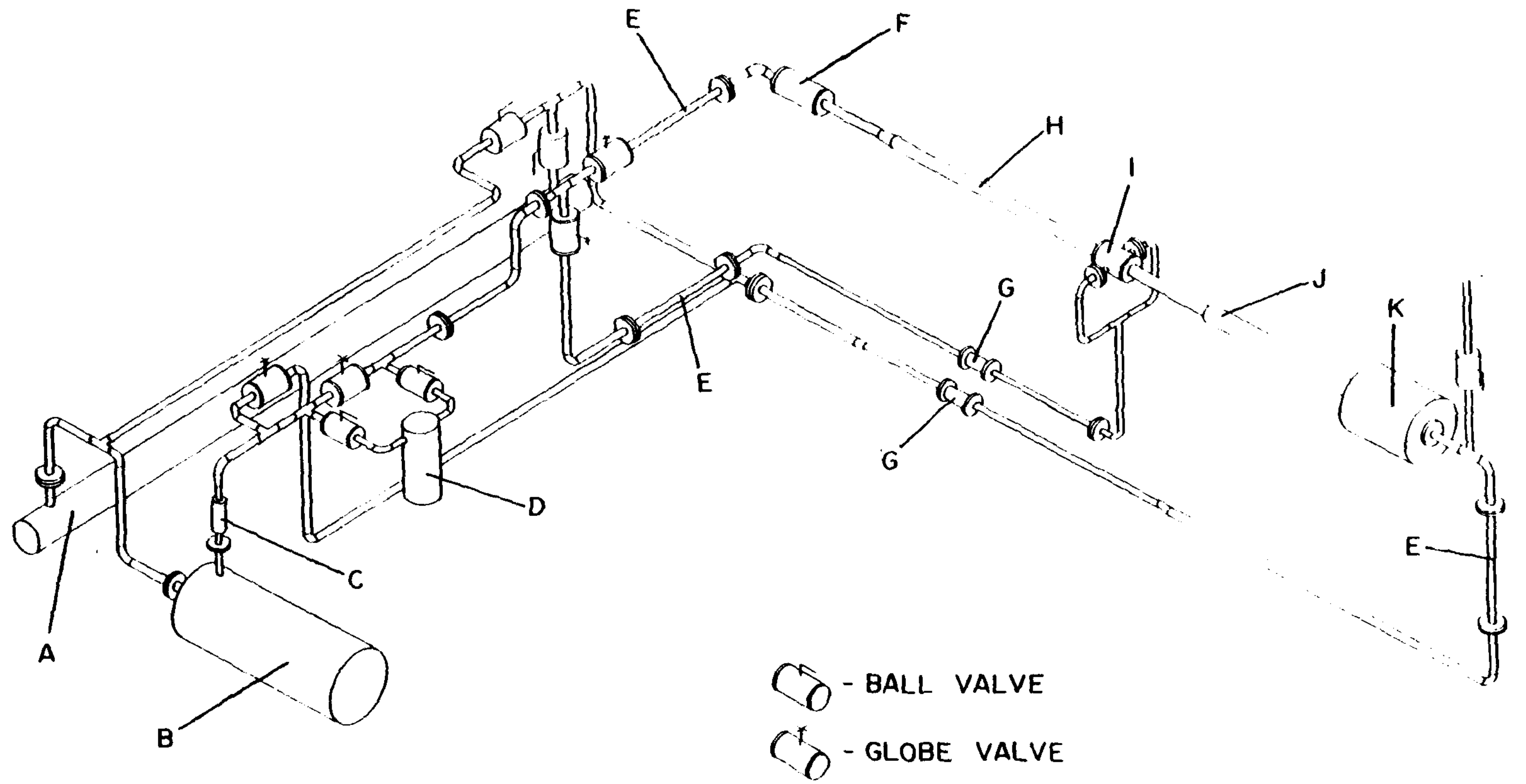
When making LDV measurements it is important that no abnormally large particles enter the sampling volume, since they may cause incorrect velocity measurements. For this reason, a 0.5 micron Filterite filter has been incorporated into the loop. The filter itself is located in a separate loop off of the main feed line to the test section. With a capacity of only 5 gpm it will be used only during the starting up period of any runs and closed off when actual measurements are being taken. This will help to insure that scale or other particles formed while the system is not in use do not interfere with the LDV measurements.

Other auxiliary loop components include a mixing plenum, flow straightener and braided, flexible piping. The mixing plenum is a 7.3 gallon cylindrical tank located at the downstream end of the test section. This plenum will provide a chamber in which the flow can uniformly mix and the final bulk temperature measured. The flow straightener is a 12'' length of 1 1/2'' pipe with stainless steel vanes dividing the cross-section of the pipe into quarters. This straightener is located upstream of the developing pipe, where it will reduce secondary flows before the water enters the test section.

Upstream of the flow straightener, downstream of the plenum, and in the tangential line to the swirl producer are 18'' lengths of braided, flexible pipe. These sections were specified to help isolate the test section from vibrations in the rest of the loop. Further vibration elimination is obtained with braided pipe bolted to the pump inlet and discharge. These precautions against vibration were included in the loop design to provide as good an environment as possible for LDV measurements.

The entire test loop is supported by a 1 1/2'' square steel tubing structure, bolted to the concrete lab floor over 1/8'' rubber pads. The loop is positioned on the support structure so that the test section centerline is at a working height of 4 feet.

A schematic of the overall loop design with legend is shown in Figure III.2. Figures III.3-III.4 are photographs of the portions of the loop which have been assembled to date.



A- HEAT EXCHANGER
B- PUMP
C- CHECK VALVE
D- FILTER

E- FLEXIBLE PIPE
F- FLOW STRAIGHTENER
G- TURBINE FLOWMETER
H- DEVELOPING TUBE

I- SWIRL GENERATOR
J- TEST SECTION
K- MIXING CHAMBER

Fig. III.2 Schematic of swirl flow test rig.

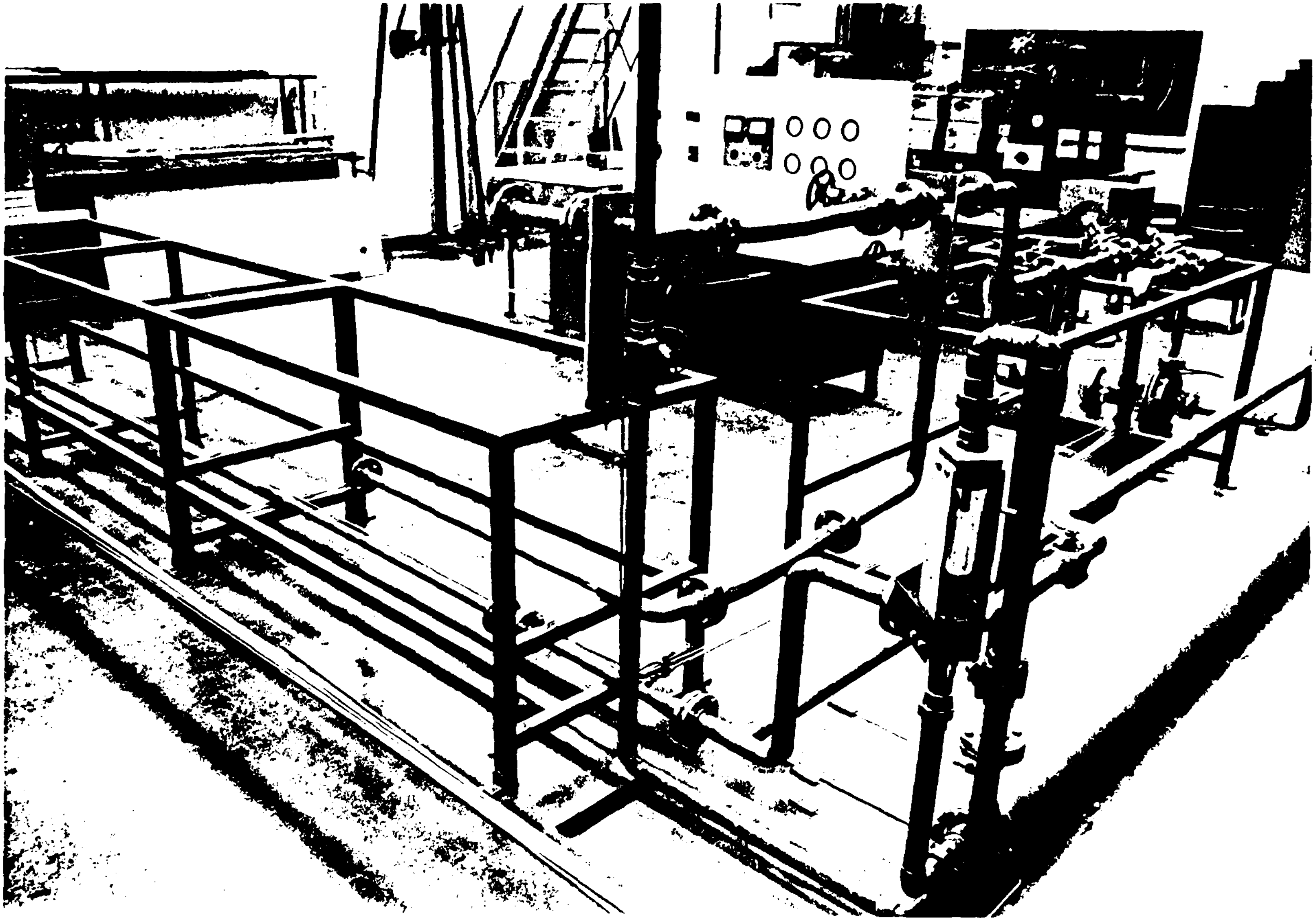


Fig. III.3 Photograph of partially constructed swirl flow test rig. View from heat exchanger side.

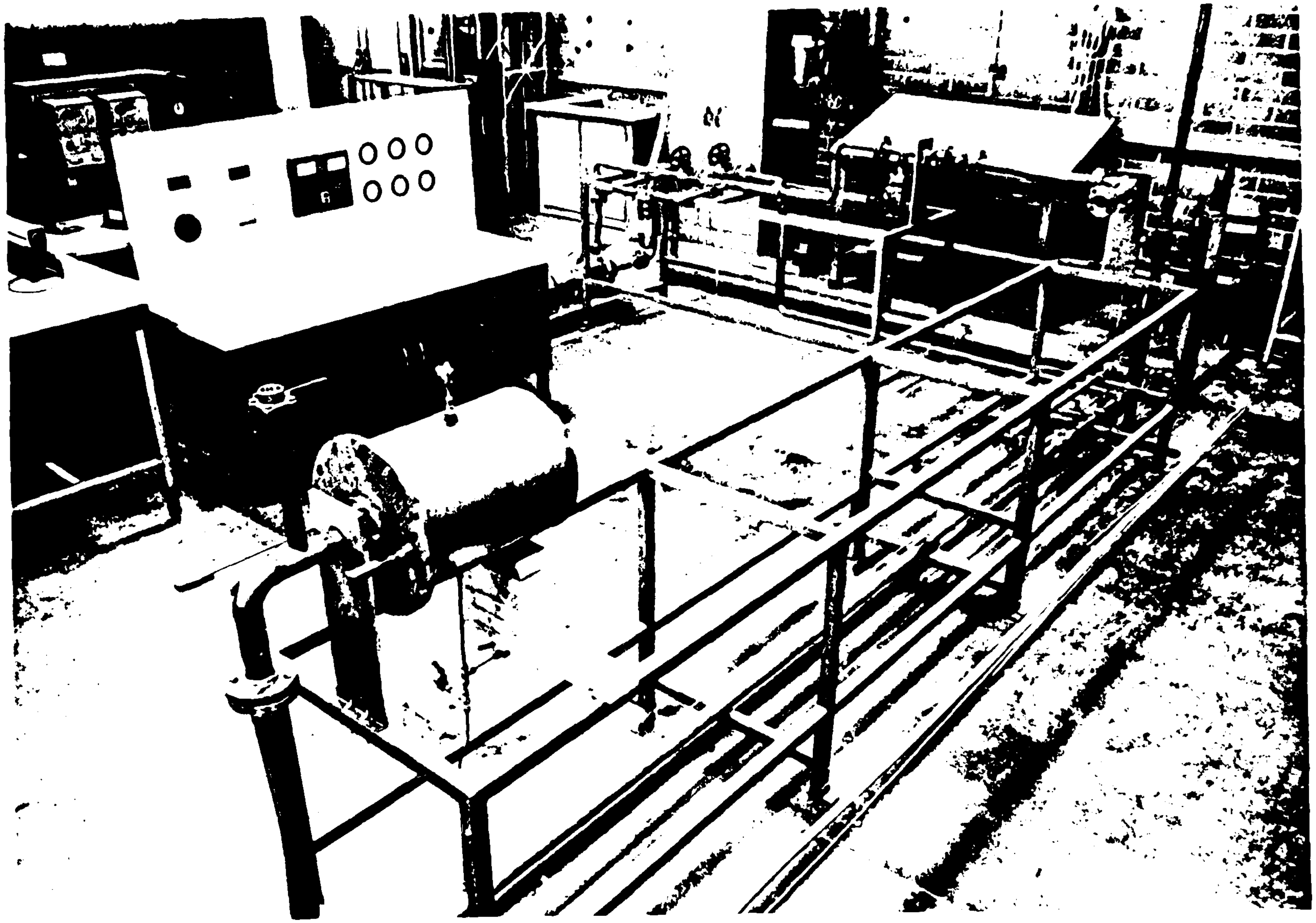


Fig. III.4 Photograph of partially constructed swirl flow test rig. View from downstream plenum.

IV. NUMERICAL MODELING

A. Introduction

Preliminary work has been carried out on various aspects of numerical modeling envisaged for the project. Efforts were directed at familiarization with the popular TEACH code together with assessment of its prediction capabilities against available published data. This was thought essential in order to rationally plan strategies for suitable modifications of the code or, alternatively, for the development of a new code.

A survey of the literature reveals a continuing lack of good experimental data on swirling flow with abrupt expansion either with or without heat transfer. A copy of a very recent thesis by Rhode [2] where such data were reported, has been ordered. However, the use of a five-hole pitot probe for the acquisition of these data renders them of dubious value. Nevertheless they are expected to offer a good starting point for comparison with model prediction for the case of swirling flow with abrupt expansion. In the meantime, prediction capability of the TEACH code has been studied against the data of Back and Roschke [3], Moon and Rudinger [4], and Chaturvedi [5]. Based on these exploratory computer runs as well as other recently published work, it becomes evident at this stage that the code has its limitations (in terms of both the numerical scheme and the embodied turbulence model) for the case of a swirling recirculating flow field.

B. The TEACH CODE

The present ASU version of TEACH was obtained in October, 1981, in the form of a card deck from Dr. M.A. Habib, who was then at the University of Arizona, Tucson. The code had been operational there on a CDC computer. After some initial debugging, the code was made operational on the ASU Amdahl system.

Unfortunately, there is no detailed documentation on the code. The report by Gosman and Ideriah [6] was written for an earlier version of the code. Consequently, an attempt has been made to generate a symbol table for the code by linking it with the underlying mathematical formulation and making use of various other pieces of published information besides the work of Gosman et al. [7] and Patankar [8].

1. Organization

The present version of the TEACH code is organized to possess the following features:

(i) Steady-state Navier-Stokes formulation in two-dimensional rectangular or axisymmetric polar coordinates.

(ii) Use of ''primitive'' variables as hydrodynamic variables.

(iii) Two-equation, k-ε model of turbulence.

(iv) Implicit, conservative formulation of the difference equations.

(v) ''Hybrid'' or ''mixed central and upwind'' difference scheme of Spalding [9].

(vi) ''Staggered-grid'' arrangement of Harlow and Amsden [10] with capability to handle non-uniform grid.

(vii) SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm of Patankar and Spalding [11].

(viii) LBL (Line-By-Line) use of TDMA (Tri-Diagonal Matrix Algorithm) for iterative solution of the finite difference equations.

(ix) User-specified but ''program-constant'' under-relaxation parameters for each dependent variable to handle non-linearity.

2. Test Runs

Developing Laminar Flow in a Straight Circular Pipe:

As an initial test run of the code, data were set up to compute laminar flow in the entrance region of a straight circular pipe. Flow Reynolds number was selected to yield a reasonable entrance length so that fully-developed exit boundary conditions were satisfied. Results of the computation were found to be in good agreement with the expected parabolic velocity profile at the exit and the estimated entrance length for flow development. However, it took a large number of iterations ($\sim$ 2000) before the solution converged to within 10^{-4}.

Developing Turbulent flow in a Straight Circular Pipe

In this case also, the code reasonably predicted the fully-developed turbulent velocity profile corresponding more or less to the 1/7th power law. The ratio of mean to center line velocity was found to be ~ 0.82. Total number of iterations to convergence within 10^{-4} was less than required for the laminar case, however the total computation time was more in this case because of the inclusion of k-ε equations of turbulence.

Comparison with the Data of Back and Roschke [3]

The experimental data pertained to water flow through an abrupt pipe expansion. Reattachment lengths were reported for Reynolds numbers between 20 and 4200. The flow geometry considered is sketched in Fig. IV.1.

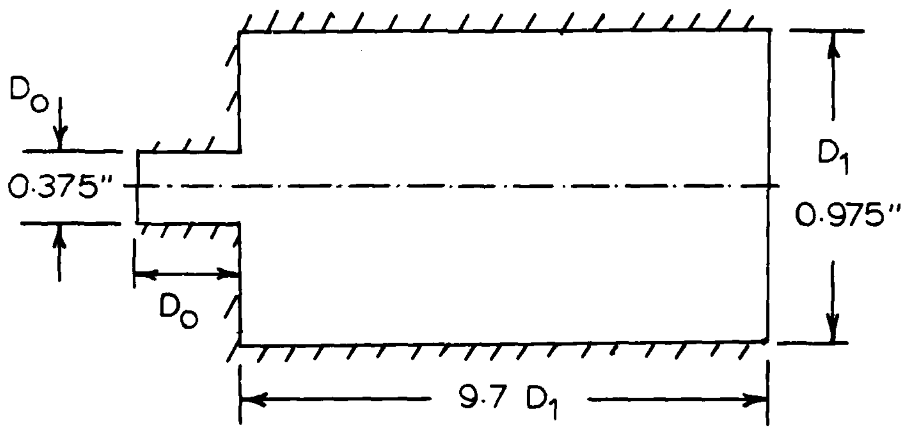


Fig. IV.1 Geometry of Back and Roschke [3]

A total of 22x22 grid points were used for the computation. The grid was taken to be nearly uniform in the radial direction and expanding in the axial direction with an expansion ratio of 1.2.

Results of our computations for different Reynolds numbers are summarized in Table IV.1 and plotted in Figs. IV.2 and IV.3.

Run No.	Re	Reattachment Length (x/h)	Remarks
1	25	1.57-2.43	Laminar
2	50	2.4-3.4	' '
3	100	5-6	' '
4	250	14-16	' '
5	500	28-30	' '
6	1000	> 32	' '
7	1000	14-16	Turbulent
8	4000	6.2-8.0	' '

(h = step height)

TABLE IV. 1 Reattachment Lengths Vs. Reynolds No.

In general, the code seems to underpredict the reattachment lengths both in the fully laminar and fully turbulent regions. The absence of this trend and the large discrepancy between experiment and prediction for Reynolds numbers between 250 and 1000 could be attributed to a transition flow region.

The TEACH code seems to have been optimally set-up for handling turbulent flow. While carrying out laminar flow computations for Reynolds numbers less than 100, the solution was found to oscillate and take a large number of iterations for convergence to within even 1%. For Re = 25, it appeared even nonconvergent. The problem was finally traced to incorrect initial guesses for the pressure field. For the turbulent case the code starts with zero pressure at all the grid points (except the point where reference pressure is set). This was verified to

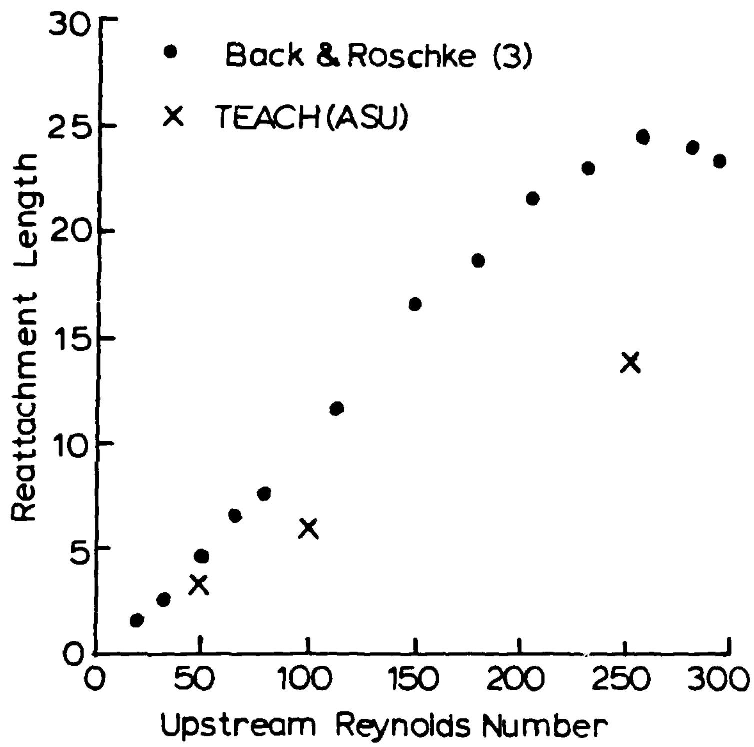


Fig. IV.2 Reattachment length comparison with the results of Back & Roschke (3) -- laminar shear layer regime.

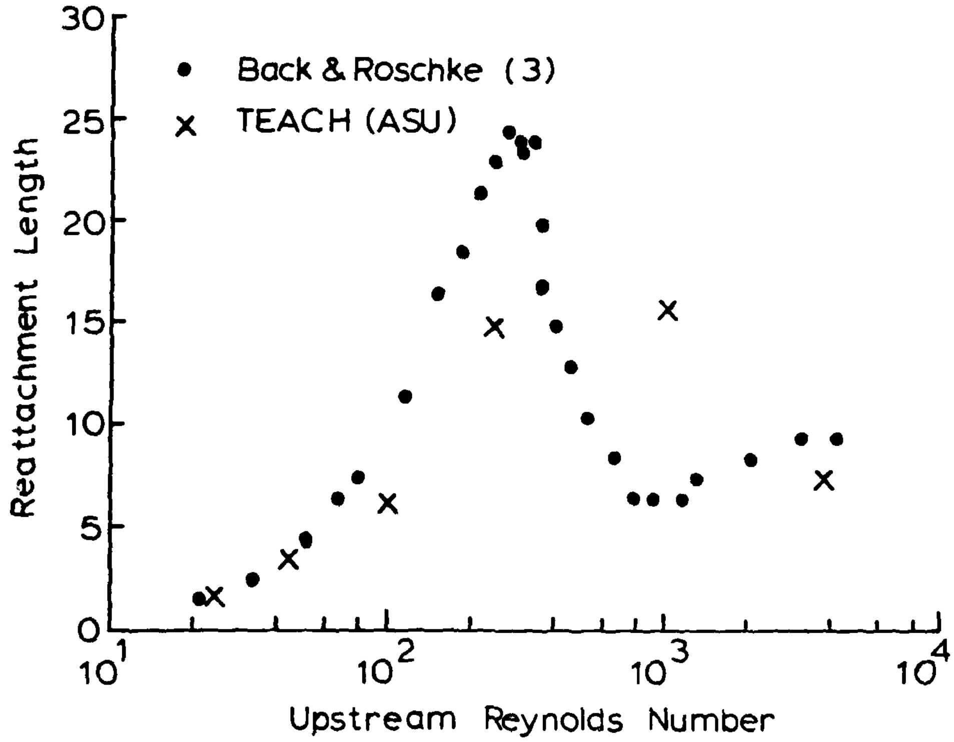


Fig. IV.3 Reattachment length comparison with the results of Back & Roschke (3).

be optimal to effect faster convergence. However in the case of laminar flow an initial guess on the pressure field using Borda-Carnot expression [12] was found to significantly improve convergence:

$$P = P_s + \frac{1}{2} \rho \, U_s^2 Cp_{bc} \qquad (1)$$

where $Cp_{bc} = 2\beta^2(1-\beta^2)$ and $\beta = \frac{D_o}{D_1}$.

In view of the short pipe entry length before the expansion, a uniform inlet velocity profile was assumed.

Comparison with the Data of Moon and Rudinger [4]

The data in this case consisted of mean axial velocities for air flow in a circular duct with sudden expansion. The geometry of flow configuration is sketched in Fig. IV.4.

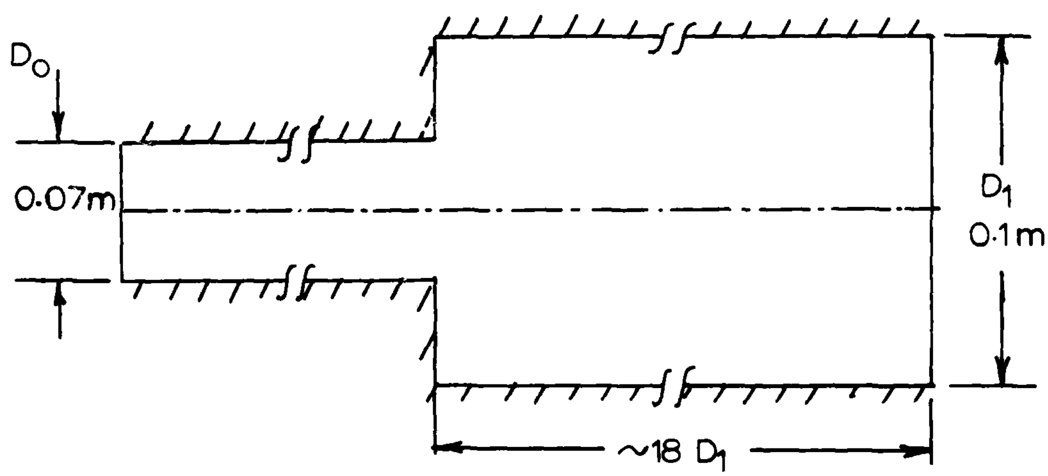


Fig. IV.4 Geometry of Moon and Rudinger [4]

The results of our computations for this case are shown plotted in Figs. IV.5 and IV.6, which also show their theoretical predictions using a different version of TEACH.

For this case, since the entry pipe prior to the sudden expansion was about 18 diameters long, a fully-developed turbulent velocity profile at the step was thought appropriate and the inlet values for k and ε were accordingly set using mixing length theory. The prediction was found to agree poorly with the reported data. However when $k_{in} = 10^{-30}$ (small) and $\varepsilon_{in} = 10^{30}$ (large) were set, the prediction remarkably improved. These values of k and ε reflect laminar like inlet conditions. However, in view of the flow Reynolds number of 2.8×10^5 such a specification on k and ε seem unreasonable.

A total of 25x15 grid points were used for the computation.

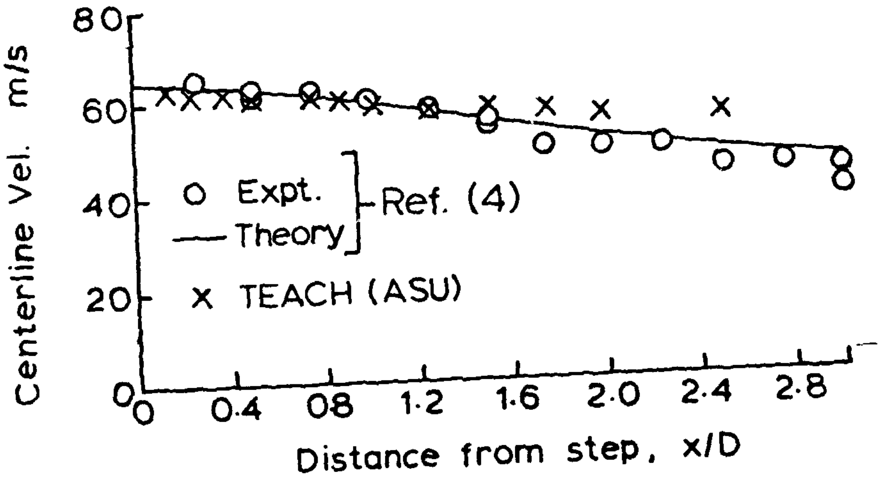


Fig. IV.5 Comparison of centerline velocity with measurements and prediction of Moon & Rudinger (4).

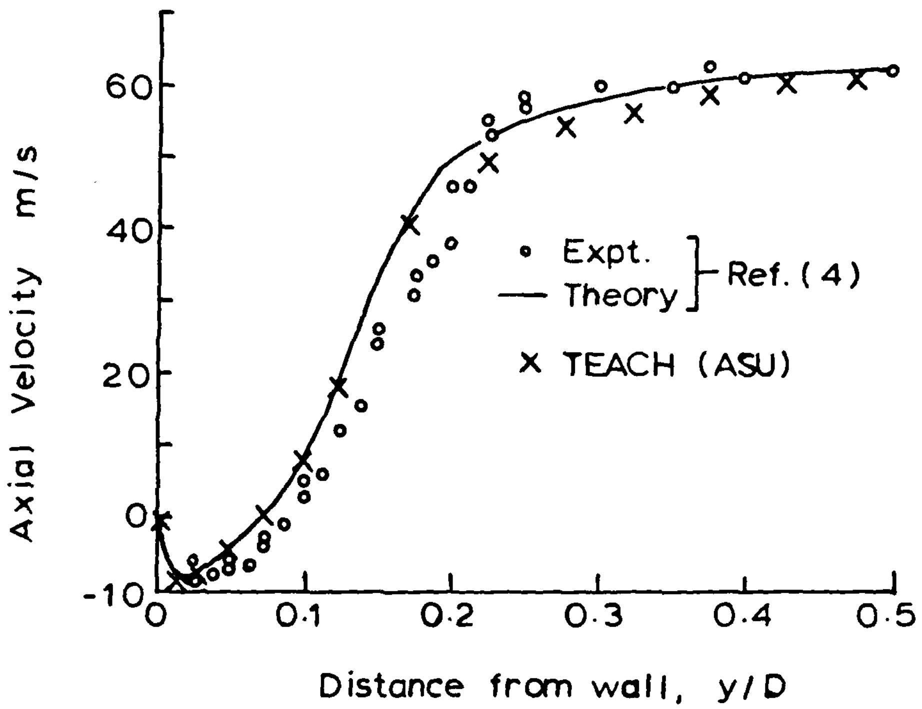


Fig. IV.6 Comparison of axial velocity profile at x/D = 0.75 with measurements and prediction of Moon & Rudinger (4).

Comparison with the Data of Chaturvedi [5]

These data sets appear to be the best available measurements of both mean flow and turbulence quantities for flow through a sudden pipe expansion. The pipe geometry used in the experiment is shown in Fig. IV. 7. The working fluid was air.

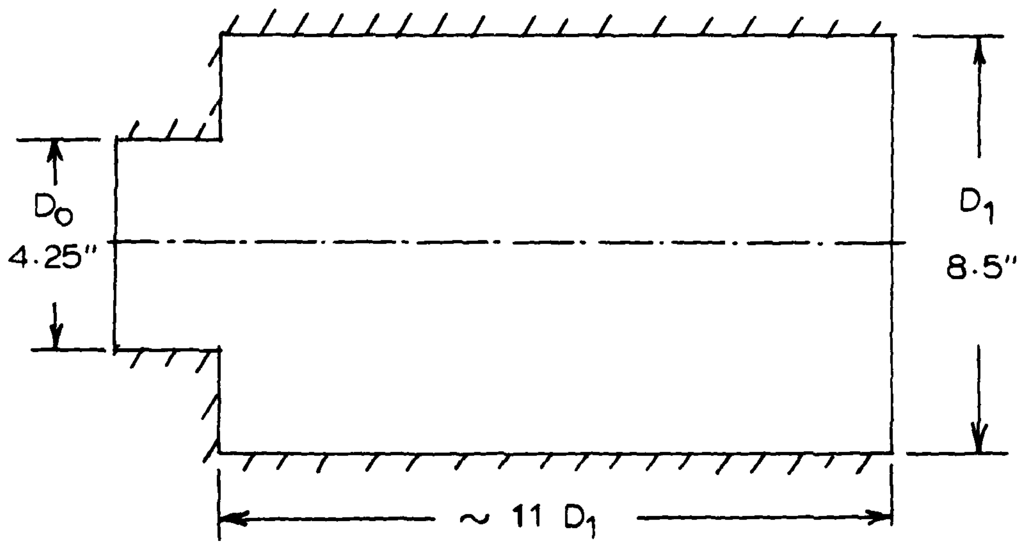


Fig. IV. 7 Geometry of Chaturvedi [5]

In view of the short entry pipe a uniform velocity profile was assumed at the expansion step, for these computations.

A large number of computer runs were made using the TEACH code in order to compare the predicted results with these experimental data. Based on this comparative study, the following conclusions may be drawn:

(i) The code predicts a reattachment length between $x/D_o \sim 4\text{-}5$ compared to the experimental value of $x/D_o \sim 6$.

(ii) The radial variation of mean axial velocity at different axial locations is reasonably predicted.

(iii) Predicted values of turbulent shear stress ($-\rho\overline{u'v'}$) are higher than experimental values although the trend at different axial locations is predicted satisfactorily.

(iv) Computed values of local turbulence intensities, $\frac{u'}{U_o}$ and $\frac{v'}{U_o}$ are in fair agreement with reported data though the discrepancies are higher in the recirculation region.

3. Observations and Preliminary Modifications

(i) A typical computer run of the TEACH Code on Amdahl takes about 5 minutes of CPU time. Recent installation of an IBM 3081 computer system will result in a reduction of this figure.

(ii) ''Number of iterations to convergence'' was found to be a complex function of both the number of grid points and their non-uniform spacings. From considerations of stability and computational times, it is recommended that the grid expansion ratio be kept within 1.5.

(iii) Input to the code required ''user-specified'' coordinates of all the grid points which is both inconvenient and prone to error. The code was therefore modified to generate a non-uniform grid based on the overall dimensions of the flow field and the specified grid expansion factors for both radial and axial coordinate directions.

(iv) The OUTPUT subroutine of the code was previously organized to give all the scalar dependent variables at the main grid points whereas velocities were printed at the staggered grid points. This was modified to have all the dependent variables printed at the main grid points. In addition, the provision was made to get output at user specified print points which could in general be different from computational grid points. Two-dimensional linear interpolation was used to compute values at the print points using those available at the computational grid points.

4. <u>Major Shortcomings of TEACH and State-of-the-Art</u>

Both the numerical scheme and k-ε model of turbulence appear to be inadequate to handle pipe flow with sudden expansion. The introduction of swirl makes matters worse as the recent experiences of Habib and Whitelaw [13-14] and Rhode and Lilley [15] indicate. The mere change of the turbulence model to either an algebraic stress model (ASM) or a full Reynolds stress model (RSM) may not be sufficient in the absence of a framework of a proper numerical scheme. Bradshaw [16] has rightly pointed out that in complex flows, numerical errors, due to the use of coarse meshes to conserve computer storage and time, take the form of an extra gradient diffusion of each transported quantity and can equal or swamp the inaccuracies of turbulence models. It therefore appears clear that efforts need to be directed towards improvements in both the numerical scheme and turbulence model, the former being a mandatory framework and the latter being highly dependent on the availability of reliable experimental data.

Numerical Scheme

The use of ''hybrid'' or ''mixed central and upwind'' differencing in the TEACH code has been the subject of major controversy over the last decade. The scheme is known to introduce ''false diffusion'' (also called ''false viscosity,'' ''numerical diffusion '' or ''artificial diffusion''). As discussed by Patankar [8], false diffusion is essentially a multidimensional phenomenon; it has absolutely no counterpart in steady one-dimensional situations. The matter of false diffusion attains importance when the grid Peclet numbers are large and the local velocity vector is even slightly skewed relative to the numerical grid lines with non-zero gradient of the dependent variable in the direction normal to the flow. Just these conditions prevail in the case of turbulent,recirculating flows. It is at least partly because of this artificial diffusion that the TEACH code tends to underpredict the reattachment length in all the flow situations considered.

During one of the computer runs on Chaturvedi's data, output was obtained for grid Peclet numbers in both axial and radial directions. It was found that in the radial direction Peclet numbers were usually < 2 implying the use of central differencing whereas in the axial direction they were ~ 100 or more. It was evident that any attempt to reduce the grid Peclet numbers in the axial direction so as to procure stable central differencing would be futile in view of the required large number of grid points.

The reduction of artificial diffusion in the computation of steady-state recirculating flows at no significant loss of computational economy has been the objective of two methods proposed recently by Raithby [17] and Leonard [18]. The ''skew-upwind-differencing scheme'' of Raithby, although formally only first-order accurate, yields a significant reduction in skewness errors by *partially simulating an upwind discretization* in a stream-line (i.e. natural) coordinate system, in which case skewness errors are entirely absent. The method of Leonard is of a more fundamental nature, designed to eliminate artificial diffusion altogether by using, in a conservative manner, quadratic, upstream-weighted interpolation for the convection terms. Current experience with both schemes, particularly with that of Leonard, is very limited.

Recent comparative studies carried out on the three schemes by Leschziner [19] and Leschziner and Rodi [20] show the inadequacy of the ''hybrid'' scheme of the TEACH code. They found that the skew-upwind scheme required roughly 50% more computing time compared to the time required by either the upwind or quadratic-interpolation schemes.

Method of Solution

The solution method used in the code is an iterative type based on line-by-line use of a tri-diagonal matrix algorithm for the set of finite-difference equations for each dependent variable. The non-linearity of the equations is handled through the application of under-relaxation parameters independently set for each dependent variable and the transport properties. This renders the approach rather problem dependent and may, in some cases, result in very slow convergence requiring a number of experimental computer runs to strike suitable values for these parameters. Another approach to introduce under-relaxation is through the use of contrived-transient discussed by deSocio et al [21].

Pressure Velocity Coupling

When the mathematical formulation is based on primitive variables, there is another problem of significant importance associated with the pressure-velocity coupling. Raithby [22] points out that this problem is mainly responsible for the slow convergence of existing solution methods. The TEACH code uses the ''SIMPLE'' algorithm of Patankar and Spalding [11] which has been subsequently revised to ''SIMPLER'' by Patankar [23]. Raithby has critically compared various available algorithms for the pressure-velocity coupling and suggested a refined algorithm called ''PUMPIN.''

Turbulence Model

As discussed earlier, an accurate framework of numerical scheme is prerequisite to the testing of various turbulence models. In the absence of such a framework, no conclusions can, in general, be drawn on the performance of any particular turbulence model. In comparisons made between computations and experimental data, errors arising from model defects cannot be separated from numerical errors. Nevertheless, there is a fundamental objection to the use of the k-ε turbulence model employed in the TEACH code when dealing with complex shear flows (e.g. recirculating flow with or without swirl) where the isotropy assumption implied by the effective scalar viscosity hypothesis is physically unrealistic. Either an algebraic stress model or a full Reynolds stress model would be more appropriate in this case, though the latter would significantly increase the computation time. Also, the TEACH code presently uses logarithmic wall functions to link the near wall grid lines to the boundaries. The use of these wall functions is based on an equilibrium flow assumption (production of $k = \varepsilon$) near the wall which is well justified. However a typical output of these quantities while using the code on Chaturvedi's data showed significant deviations from the equilibrium assumption. Recently Spalding and Elhadidy [24] have proposed another approach as an alternative to the use of logarithmic wall functions and it appears worth pursuing.

C. <u>Future Action</u>

(i) Carry out further test runs using the TEACH code in order to compare with the experimental data of Weske and Sturov [25] on turbulent swirled flows in a straight circular pipe and also with the experimental data of Rhode [2] for the case of swirling flow with abrupt pipe expansion.

(ii) Repeat the computer runs under 1 with the TEACH code modified to include an algebraic stress turbulence model.

(iii) Repeat the complete set of test runs on available data using a modified version of the computer code used by Neitzel and Davis [26] which is based on the Predictor Corrector Multiple Iteration (PCMI) technique [27]. The code would be modified to include an algebraic stress model of turbulence.

(iv) At the end of above study, a suitable strategy would be worked out for modification of available codes as well as development of a new code if found necessary.

References

[1] Bicen, A. F., ''Refraction Correction for LDA Measurements in Flows with Curved Optical Boundaries,'' TSI Quart., 8, no. 2, 1982, 10.

[2] Rhode, D. L., ''Prediction and Measurements of Isothermal Flow fields in Axisymmetric Combustor Geometries,'' Ph.D. Thesis, Oklahoma State University, Stillwater, OK, Dec. 1981.

[3] Back, L. M., and Roschke, E. T., ''Shear-Layer Flow Regimes and Wave Instabilities and Reattachement Lengths Downstream of an Abrupt Circular Channel Expansion,'' J. Appl. Mech., Sept. 1972, 677.

[4] Moon, L. F., and Rudinger, G., ''Velocity Distribution in an Abruptly Expanding Circular Duct,'' J. Fluids Eng., Mar. 1977, 226.

[5] Chaturvedi, M. C., ''Flow Characteristics of Axisymmetric Expansions,'' ASCE, J. Hydraulics Div., May 1963, 61.

[6] Gosman, A. D., and Ideriah, F. J. K., ''TEACH: A General Computer Program for Two-Dimensional Turbulent, Recirculating Flows,'' June 1976, Dept. of Mech. Eng., Imperial College, London, S.W. 7.

[7] Gosman, A. D., Pun, W. M., Runchal, A. K., Spalding, D. B., and Wolfshtein, M., <u>Heat and Mass Transfer in Recirculating Flows</u>, Academic Press, London, 1969.

[8] Patankar, S. V., <u>Numerical Heat Transfer and Fluid Flow</u>, McGraw-Hill, New York, 1980.

[9] Spalding, D. B., ''A Novel Finite-Difference Formulation for Differential Expressions Involving both First and Second Derivatives,'' Int. J. Num. Meth. Eng., 4, 1972, 551.

[10] Harlow, F. M. and Amsden, A. A., ''Numerical Calculation of Almost Incompressible Flow,'' J. Comp. Phys. 3, 1968, 1.

[11] Patankar, S. V., and Spalding, D. B., ''A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows,'' Int. J. Heat Mass Trans., 15, 1972, 1787.

[12] Johnston, J. P. ''Internal Flows,'' in <u>Turbulence</u>, P. Bradshaw, ed., Vol. 12, Topics in Applied Physics, Springer-Verlag, Berlin, 1978.

[13] Habib, M. A. and Whitelaw, J. H. ''Velocity Characteristics of a Confined Co-axial Jet,'' J. Fluids Eng., 101, 1979, 521.

[14] Habib, M. A., and Whitelaw, J. M., ''Velocity Characteristics of Confined Coaxial Jets with and without Swirl,'' J. Fluids Eng., 102, 1980, 47.

[15] Rhode, D. L. and Lilley, D. G., ''Mean Flow Fields in Axisymmetric Combustor Geometrics with Swirl,'' paper No. AIAA-82-0177., AIAA 20th Aerospace Science Meeting, Jan. 11-14, 1982, Orlando, FL.

[16] Bradshaw, P., ''Fundamentals and Applications of Turbulent Flow,'' Prince Distinguished Lecture Series, Dept. of Mechanical and Energy Systems Eng., ASU, Tempe, Arizona, 1982.

[17] Raithby, G. D., ''Skew Upwind Differencing Schemes for Problems Involving Fluid Flow,'' Comp. Meths. Appl. Mech. Eng. 9, 1976, 153.

[18] Leonard, B. P., ''News-Flash: Upstream Parabolic Interpolation,'' Proceedings of 2nd GAMM Conference on Numerical Methods in Fluid Mechanics, Koln, Germany, 1977, 97.

[19] Leschziner, M. A., ''Practical Evaluation of Three Finite Difference Schemes for the Computation of Steady-State Recirculating Flows,'' Comp. Meth. Appl. Mech. Eng. 23, 1980, 293.

[20] Leschziner, M. A. and Rodi, W., ''Calculation of Annular and Twin Parallel Jets Using Various Discretization Schemes and Turbulence Model Variations,'' J. Fluids Eng., 103, 352.

[21] deSocio, L. M., Sparrow, E. M., and Eckert, E.R.G., ''The Contrived Transient-Explicit Method for Solving Steady-State Flows: Application to a Rotating, Recirculating Flow,'' Computers and Fluids, 1, 1973, 273.

[22] Raithby, G. D. et. al., ''The Investigation of Numerical Modeling Techniques for Recirculating Flows. Part II on Economical Numerical Techniques,'' EPRI CS-1665, Part 1,2, Feb. 1981.

[23] Patankar, S. V., ''A Calculation Procedure for Two-Dimensional Elliptic Situations,'' Num. Heat Transfer, 2, 1979, .

[24] Spalding, D. B. and Elhadidy, M. A., ''New Approach to Calculating, Turbulent Flows in the Near-Wall Region,'' HTS/79/9, Aug. 1979.

[25] Weske, D. R. and Sturov, G. Ye, ''Experimental Study of Turbulent Swirled Flows in a Cylindrical Tube,'' Fluid Mech. Soviet Research, 3, 1974, 77.

[26] Neitzel, G. P. and Davis, S. H., ''Centrifugal Instabilities During Spin-down to Rest in Finite Cylinders. Numerical Experiments,'' J. Fluid Mech. 102, 1981, 329.

[27] Rubin, S. G. and Lin, T. C., ''A Numerical Method for Three-dimensional Viscous Flow--Applications to the Hypersonic Leading Edge,'' J. Comp. Phys., 9, 1972, 339.

DISTRIBUTION LIST

HEAT TRANSFER

	One Copy except as noted
Mr. M. Keith Ellingsworth Mechanics Division Office of Naval Research 800 N. Quincy Street Arlington, VA 22203	2
Defense Documentation Center Building 5, Cameron Station Alexandria, VA 22314	12
Technical Information Division Naval Research Laboratory 4555 Overlook Avenue SW Washington, DC 20375	6
Professor Paul Marto Department of Mechanical Engineering U.S. Naval Post Graduate School Monterey, CA 93940	
Professor Bruce Rankin Naval Systems Engineering US Naval Academy Annapolis, MD 21402	
Mr. Doug Marron Code 05R13 Crystal Plaza #6 Naval Sea Systems Command Washington, DC 20362	
Steam Generators Branch, Code 5222 National Center #4 Naval Sea Systems Command Washington, D.C. 20362	
Heat Exchanger Branch, Code 5223 National Center #3 Naval Sea Systems Command Washington, D.C. 20362	

Mr. Ed Ruggiero, NAVSEA 08
National Center #2
Washington, D.C. 20362

Dr. Earl Quandt Jr., Code 272
David Taylor Naval Ship R&D Center
Annapolis, MD 21402

Mr. Wayne Adamson, Code 2722
David Taylor Naval Ship R&D Center
Annapolis, MD 21302

Dr. Win Aung
Heat Transfer Program
National Science Foundation
Washington, DC 20550

Mr. Michael Perlsweig
Department of Energy
Mail Station E-178
Washington, DC 20545

Dr. W.H. Theilbahr
Chief, Energy Conservation Branch
Dept. of Energy, Idaho Operations Office
550 Second Street
Idaho Falls, Idaho 83401

Professor Ephriam M. Sparrow
Department of Mechanical Engineering
University of Minnesota
Minneapolis, MN 55455

Professor S.V. Patankar
Department of Mechanical Engineering
University of Minnesota
Minneapolis MN 55455

Professor Daryl Metzger
Chairman, Mechanical and
Energy Systems Engineering
Arizona State University
Tempe AZ 85281

Professor Ronald So
Mechanical and Energy Systems
Engineering
Arizona State University
Tempe AZ 85281

Professor J.A.C. Humphrey
Department of Mechanical Engineering
University of California, Berkeley
Berkeley, CA 94720

Professor Brian Launder
Thermodynamics and Fluid Mechanics Division
University of Manchester
Institute of Science & Technology
PO88 Sackville Street
Manchester M601QD England

Professor Shi-Chune Yao
Department of Mechanical Engineering
Carnegie-Mellon University
Pittsburgh, PA 15213

Professor Charles B. Watkins
Chairman, Mechanical Engineering Department
Howard University
Washington, DC 20059

Professor Adrian Bejan
Department of Mechanical Engineering
University of Colorado
Boulder, CO 80309

Professor Donald M. McEligot
Department of Aerospace and Mechanical Engineering
Engineering Experiment Station
University of Arizona 85721

Professor Paul A. Libby
Department of Applied Mechanics and Engineering Sciences
University of California San Diego
Post Office Box 109
La Jolla, CA 92037

Professor C. Forbes Dewey, Jr.
Fluid Mechanics Laboratory
Massachusetts Institute of Technology
Cambridge, MA 02139

Professor William G. Characklis
Dept. of Civil Engineering and Engineering Mechanics
Montana State University
Bozeman, MT 59717

Professor Ralph Webb
Department of Mechanical Engineering
Pennsylvania State University
208 Mechanical Engineering Bldg.
University Park, PA 16802

Professor Warren Rohsenow
Mechanical Engineering Department
Massachusetts Institute of Technology
77 Massachusetts Avenue
Cambridge, MASS 02139

Professor A. Louis London
Mechanical Engineering Department
Bldg. 500, Room 501B
Stanford University
Stanford, CA 94305

Professor James G. Knudsen
Associate Dean, School of Engineering
Oregon State University
219 Covell Hall
Corvallis, Oregon 97331
Professor Arthur E. Bergles
Mechanical Engineering Department
Iowa State University
Ames, Iowa 50011

Professor Kenneth J. Bell
School of Chemical Engineering
Oklahoma State University
Stillwater, Oklahoma 74074

Dr. James Lorenz
Component Technology Division
Argonne National Laboratory
9700 South Cass Avenue
Argonne, Illinois 60439

Dr. David M. Eissenberg
Oak Ridge National Laboratory
P.O. Box Y, Bldg. 9204-1, MS-0
Oak Ridge, Tennessee 37830

Dr. Jerry Taborek
Technical Director
Heat Transfer Research Institute
1000 South Fremont Avenue
Alhambra, CA 91802

Dr. Simion Kuo
Chief, Energy Systems
Energy Research Laboratory
United Technology Research Center
East Hartford, CT 06108

Mr. Jack Yampolsky
General Atomic Company
P.O. Box 81608
San Diego, CA 92138

Mr. Ted Carnavos
Noranda Metal Industries, Inc.
Prospect Drive
Newtown, CONN 06470

Dr. Ramesh K. Shah
Harrison Radiator Division
General Motors Corporation
Lockport, New York 14094

Dr. Ravi K. Sakhuja
Manager, Advanced Programs
Thermo Electron Corporation
101 First Avenue
Waltham, MASS 02154

Mr. Robert W. Perkins
Turbotec Products, Inc.
533 Downey Drive
New Britian, CONN 06051

Dr. Keith E. Starner
York Division, Borg-Warner Corp.
P.O. Box 1592
York, PA 17405

Mr. Peter Wishart
C-E Power Systems
Combustion Engineering, Inc.
Windsor, CONN 06095

Mr. Henry W. Braum
Manager, Condenser Engineering Department
Delaval
Front Street
Florence, New Jersey 08518

Dr. Thomas Rabas
Steam Turbine-Generator Technical Operations Division
Westinghouse Electric Corporation
Lester Branch
P.O. Box 9175 N2
Philadelphia, PA 19113